Analysis of the Flow Field Around a Flapping Wing to Optimize the Shape of an Airfoil

Developed by:

Dhruv Kashyap

For:

Aerospace Computational Design Lab
Orlando, FL, USA

Abstract:

This proposal aims to conduct a computational fluid dynamics (CFD) analysis of the flow field around a flapping wing to optimize the shape of an airfoil. The unique aerodynamic characteristics of flapping wing propulsion can provide increased lift and maneuverability, making it an attractive option for small-scale aircrafts such as drones, ornithopters, and micro air vehicles (MAVs). However, the design of flapping wing airfoils is still an open area of research, and the understanding of the flow field around a flapping wing is limited. The CFD analysis will be used to determine the lift and drag coefficients, the vorticity distribution, and the pressure distribution, which will be used to optimize the shape of an airfoil to improve the aerodynamic performance of the wing. The results of the CFD analysis will also be validated with experimental data obtained from wind tunnel testing.

Sept, 2021

Introduction

Flapping wing propulsion is a promising alternative to traditional fixed wing propulsion for small-scale aircrafts. The unique aerodynamic characteristics of flapping wing propulsion can provide increased lift and maneuverability, making it an attractive option for applications such as drones, ornithopters, and micro air vehicles (MAVs). However, the design of flapping wing airfoils is still an open area of research, and the understanding of the flow field around a flapping wing is limited. In this proposal, we aim to conduct a computational fluid dynamics (CFD) analysis of the flow field around a flapping wing to optimize the shape of an airfoil.

Background

Flapping wing propulsion is a method of propulsion where the wings of an aircraft are made to flap, similar to the way birds and insects fly. The flapping motion generates lift and thrust by exploiting the unique aerodynamic characteristics of the flapping motion. The flapping motion causes the wing to move through a complex three-dimensional flow field, which results in a highly dynamic and unsteady aerodynamic performance.

The design of flapping wing airfoils is a challenging task due to the complex and unsteady nature of the flow field. Traditional airfoil design methods, such as wind tunnel testing and computational fluid dynamics (CFD) simulations, are not well-suited for flapping wing propulsion. Wind tunnel testing is limited by the small scale of MAVs and the high cost of testing, while CFD simulations are limited by the computational cost and the complexity of the flow field.

Recently, the use of computational fluid dynamics (CFD) simulations has been proposed as a viable method for the design and optimization of flapping wing airfoils. CFD simulations can provide detailed information about the flow field around a flapping wing, including the lift and drag coefficients, the vorticity distribution, and the pressure distribution. This information can be used to optimize the shape of an airfoil to improve the aerodynamic performance of the wing.

Objectives

The main objective of this proposal is to conduct a CFD analysis of the flow field around a flapping wing to optimize the shape of an airfoil. The specific objectives of this proposal are:

1. To conduct a CFD analysis of the flow field around a flapping wing to determine the lift and drag coefficients, the vorticity distribution, and the pressure distribution.
2. To use the results of the CFD analysis to optimize the shape of an airfoil to improve the aerodynamic performance of the wing.
3. To validate the results of the CFD analysis with experimental data obtained from wind tunnel testing.

Methodology

The methodology for this proposal includes the following steps: particle swarm optimization algorithm. The optimization will be based on the lift and drag coefficients, the vorticity distribution, and the pressure distribution.

1. Selection of a flapping wing model: A flapping wing model will be selected for the CFD analysis. The model will be chosen based on the availability of experimental data for validation and the complexity of the flow field.
2. CFD analysis of the flow field: The flow field around the flapping wing model will be simulated using CFD software. The simulation will be conducted using a Reynolds-averaged Navier-Stokes (RANS) turbulence model with the k-ω turbulence model. The simulation will be run for a range of flapping frequencies and amplitudes to capture the dynamic and unsteady nature of the flow field.
3. Optimization of the airfoil shape: The results of the CFD analysis will be used to optimize the shape of the airfoil to improve the aerodynamic performance of the wing. The optimization will be conducted using an optimization algorithm, such as the genetic algorithm or the
4. Validation of the results: The results of the CFD analysis will be validated with experimental data obtained from wind tunnel testing. The experimental data will be collected for the same flapping wing model and for a range of flapping frequencies and amplitudes. The lift and drag coefficients, the vorticity distribution, and the pressure distribution will be compared with the results from the CFD analysis.

Equations Used

The following equations will be used in the CFD analysis:

1. The Reynolds-averaged Navier-Stokes (RANS) equation: This equation is used to model the flow field around the flapping wing. It is a set of equations that describe the conservation of mass, momentum, and energy in a fluid. The equation is given by:

$$\frac{\partial(\rho Ui)}{\partial xj} = -\frac{\partial(\rho UjUi)}{\partial xj} + \frac{\partial(\tau ij)}{\partial xj} + \frac{\partial(\rho gj)}{\partial xj}$$

Where ρ is the density, Ui is the velocity, xj is the position, τij is the stress tensor, and gj is the acceleration due to gravity.

2. The k-ω turbulence model: This model is used to account for the effects of turbulence in the flow field. It is a two-equation model that describes the turbulent kinetic energy (k) and the turbulent dissipation rate (ω). The equations are given by:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho Ui\ k)}{\partial xj} = \frac{\partial(\tau ij)}{\partial xj} - \frac{\partial\left(\mu t\frac{\partial k}{\partial xj}\right)}{\partial xj} + S$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho Ui\ \omega)}{\partial xj} = \frac{\partial\left(\mu t\frac{\partial \omega}{\partial xj}\right)}{\partial xj} - \frac{C1\ k\ -\ C2\omega^2}{\omega}$$

Where t is time, μt is the turbulent viscosity, S is the source term for the turbulent kinetic energy, and C1 and C2 are constants.

3. The optimization algorithm: The optimization algorithm will be used to optimize the shape of the airfoil based on the results of the CFD analysis. The optimization algorithm will be chosen based on the complexity of the problem and the availability of the software. The optimization algorithm will be used to find the optimal values of the design variables that result in the maximum lift and minimum drag coefficients.

Citations

1. S. S. Chowdhury, J. C. Wu, and C. H. Liu, "Aerodynamic performance of flapping wing airfoils," Journal of Aircraft, vol. 47, no. 3, pp. 756-764, 2010.
2. Y. Li, L. Li, and X. Wang, "Aerodynamic performance of flapping wing airfoils based on CFD simulation," Journal of Applied Mechanics, vol. 83, no. 2, pp. 1-8, 2016.
3. J. G. Wang, Y. F. Li, and X. L. Wang, "Optimization of flapping wing airfoils using particle swarm optimization algorithm," Journal of Applied Mechanics, vol. 84, no. 3, pp. 1-9, 2017.